
ADCIRC Mesh Editing and Parameters

This workshop concentrates on the tools in *SMS* that are useful for editing a finite element mesh and setting up the parameters for and *ADCIRC* analysis. You will use the following files from the Lesson 4 directory:


- `shin.grd`

4.1 Checking your Mesh

Read the file *shin.grd* into *SMS*. This file represents an *ADCIRC* grid, roughly equivalent to the one generated at the end of Lesson 3. Before using this grid, you need to make sure it is a high quality mesh. Poor quality elements will result in an unstable numerical solution. Several tools in *SMS* can help you ensure that the mesh is good.

4.1.1 Mesh Quality

SMS provides a tool to perform geometric quality checks on the elements in the grid. To view the checks that apply to *ADCIRC*:

1. Click on the *Display Options*  macro.
2. In the *2D Mesh* tab, turn *Mesh quality* on, and click on the *Options* button to the side of Mesh quality.
3. Turn on the following checks that apply to *ADCIRC*.

Check	Value	Comment
Minimum angle	On	Detects long skinny triangles
Maximum angle	On	Detects potentially long triangles
Concave quads	Off	ADCIRC uses only triangles
Maximum slope	Off	Not important to ADCIRC
Element area change	On	Large changes in area are numerically dangerous
Connecting Elements	On - 7	Detects potentially unstable areas (wagon wheels)
Ambiguous gradient	Off	Applies to quadrilateral elements – not ADCIRC
Display Legend	On	

4. Click *OK* to exit the *Element Quality* and *Display Options* dialogs.

The mesh will redraw showing that there are very few detected problems with this grid. There are a few blue lines indicating that adjacent elements with significant change in area. You will also see crosses at nodes where more than 7 elements come together. The default value for these wagon wheels is 8, you were told to set it to 7 to see how the node is identified.


4.1.2 Editing Individual Elements

When you want to clean up small areas of the mesh manually, to remove large changes in area or change connectivity where too many elements converge on node you will use three approaches in *SMS*. These include:

- Drag a node to make a small element bigger and a big element smaller.
- Insert a new node into the mesh in a big element.
- Swap the edges between two triangles.


Dragging a Node

Nodes are locked by default in *SMS* to prevent the user from accidentally moving them around. The elevation value associated with a node is really associated with the x and y location of that node, so moving a node very much is discouraged. Doing so would cause the numerical model to be inconsistent with the physical conditions you are representing. If you desire to drag a node a significant distance, should reinterpolate the elevation data to the grid. Try dragging nodes to eliminate some of the area changes warning:


1. Select Nodes | Locked to remove the check mark next to the item. This unlocks nodes.
2. Make sure the *Select Node Tool*  is active.
3. Zoom into an area with a blue line indicating an area change warning. Drag the nodes to either make the big element smaller, or the small element larger. Don't drag any nodes very far. Clean up five or six of the area change problems and then go on.

Alternatively, you can select the node and enter new (x,y) coordinate for the point in the edit window.

Insert a New Node

When the *Create Node* tool  is active, and you click in the graphics window, *SMS* will create a new node at the location under the mouse pointer. The elevation value for this point will be interpolated from the existing element that occupied the space where the node was created. This actually creates two new elements in the mesh. This tool can be used to split a large element next to smaller elements.

Swap edge


When the mesh quality check detects many elements converging on a node, the *Swap Edge*  tool can be used to adjust the element connectivity to remove the problem.

With the *Swap Edge* tool, the common edge of two adjacent triangular elements can be swapped.

Find and fix a few of the wagon wheel mesh quality problems detected by *SMS*.

4.1.3 Renumbering

This mesh has been constructed for *ADCIRC*, but before it is ready to run, the mesh nodes must be renumbered to minimize the bandwidth of the mesh. This allows the *ADCIRC* model to run efficiently. To do this:

1. Select the Select Nodestring  tool from the Toolbox and select the nodestring along the ocean boundary.
2. Choose Nodestrings | Renumber.
3. Make sure the Band width option is selected and click the OK button.

The nodes have now been renumbered for the entire mesh starting with those along the ocean boundary. This ensures that nodes are consistently numbered and allows for more efficient solution.

4.1.4 Converting to Lat/Lon

ADCIRC is more stable when the coordinates in the grid file are in geographic coordinates. If your grid was constructed in a Cartesian space, such as state plane, it should also be converted to Latitude/Longitude. Since this model was computed in New York State Plane, it should be converted. This conversion was also done at the

end of Lesson 3, but is included here because the grid used in this Lesson was saved in state plane coordinates. To convert to Geographic coordinates:

1. Select *Edit | Coordinate Conversions*.
2. Tell SMS what the current system is by clicking on the *Current Options* button.
3. Enter *State Plane NAD 83* as the horizontal coordinate system, and *New York Long Island 3104* as the zone. Make sure all the units are meters and click *OK*.
4. In the *Convert to* section, switch the *Horizontal System* to *Geographic NAD 83 (US)*.
5. Click the *Convert* button.

The geometric portion of the grid is now ready to be run in *ADCIRC*.

4.2 Building the *ADCIRC* Control File

The control file specifies values corresponding to different parameters used by *ADCIRC*. These parameters include specifications for tidal forcing, selection of terms to include, hot start options, model timing, numerical settings, and output control.

4.2.1 Main Model Control Screen

To set up the model control for *ADCIRC*, select *ADCIRC | Model Control*. This brings up the *ADCIRC Model Control* dialog. Make sure the data entries in this dialog are:

Parameter	Value	Comment
Project Title	Shinnecock	helps you identify the run
Run ID	Tides 11/2/98-12/3/98	
Non Fatal Error Override	On	keeps ADCIRC running if minor errors
Abbreviated Output	On	shortens length of text output file
Echo Screen	On	turns on screen messages
Initial Values	Cold Start	ADCIRC ramps up from trivial solution
Coordinate System	Spherical	ADCIRC works in geographic coords
Gravity	9.81	
Model Type	2D DI	Two dimensional depth integrated
Finite Amplitude Terms	On	
Wetting/Drying	On	
Advective Terms	On	

Time Derivative Terms	On	
Wave Continuity	0.01	
Lateral Viscosity	3.0	
Min Angle Tangential	100	
Generate Hotstart File	On	ADCIRC will generate a hotstart file
# Time Steps	3600	Hotstart file written every 3600 timesteps
Coriolis Option	Variable	
Model Center Latitude	39.984	Find center button will fill in
Model Center Longitude	-72.173	

Next we need to set the options for wetting and drying. To do this:

1. Click the Options... button beside the Wetting/Drying option.
2. Enter the following values in the Wetting/Drying Parameters dialog:
 - *Minimum Water Depth*..... 0.02
 - *Minimum # of Dry Timesteps*..... 10
 - *Number of Rewetting Timesteps*..... 10
 - *Minimum Velocity for Wetting*..... 0.05
3. Click OK to return to the *Model Control* dialog.

The upper right corner of the *ADCIRC Model Control Dialog* consists of several buttons to edit other input parameters for the model. These parameters are just as important as the others on this dialog. They are on other dialogs simply because of the number of parameters involved.

4.2.2 Station Output

ADCIRC normally runs on a timestep of between 1 and 5 seconds. If the data for the entire grid was saved for each timestep, the output files would be enormous. Therefore, data is saved globally at a less frequent interval. However, the model includes the capability to output more detailed information at specific locations. These locations are referred to as “Recording Stations” in *ADCIRC*. In SMS you specify these locations at node points. To illustrate the process, close the *ADCIRC Model Control Dialog* and create a recording station in the center of the inlet. To do this:

1. Select the node(s) in the center of the inlet (where detailed data is desired).
2. Select *ADCIRC | Assign BC ...*
3. Select *Elevation* and *Velocity* as the types of data you would like recorded at this (these) locations.
4. Click OK.

Now reopen the *Model Control Dialog* and select the *Station Output* button. The dialog that appears allows you to control what data is saved at the recording stations.

Since we don't have any *Concentration* or *Meteorological* station, leave them set to *No Output*. For *Elevation* and *Velocity* stations, set the parameters as shown in the following table:

Parameter	Value	Comment
Format	ASCII	
File	Use existing unit ##	Allows data from multiple runs to be added
Recording Start Day	1.0	Skip first day – ramp period
Recording End Day	32.0	
Output Every	200	Save data every 5 minutes (200 * 1.5)

The time options controlled in this dialog are also accessible through the time control dialog. The file and format options must be set here.

4.2.3 Global Output Control

The global output dialog allows you to control how often the data for the entire grid is saved. Leave *Concentration* and *Wind Stress* set to *No Output* since they are not used in this lesson. For *Elevation* and *Velocity*, set the values to:

Parameter	Value	Comment
Format	ASCII	
File	Use existing unit ##	Allows data from multiple runs to be added
Recording Start Day	0.0	
Recording End Day	32.0	
Output Every	1200	Save data every 30 minutes

(Note: in SMS 8.1, the frequency values will be specified as hours instead of timesteps)

4.2.4 Wind Control

The Wind Control dialog allows the user to specify what winds are to be used by *ADCIRC*. In this lesson we are not using wind, so leave the selection as *None*.

4.2.5 Time Control

The *Time Control* dialog allows the user to set the duration of the run, as well as all output frequency values. To set these values for this simulation:

1. Click the *Time Control* button.
2. Set the following values:
 - *Ramp Function Value:* 1.0 days
 - *Reference Time:* 0.0 days
 - *Wave Eq. Weight Factors:* 0.35, 0.3, 0.35
 - *Start Day:* 0.0 days
 - *Time Step:* 1.5 seconds

- *Run Time:* 32.0 days
- *Specify Run Time*

ADCIRC will generate two global output files, water-surface elevation and velocity. These values were set in the *Global Output* dialog. To view and edit the setting in this dialog select the *Constituent* and scroll down until *Global Elevation* is highlighted. The output times for this constituent are shown in the edit boxes below the *Constituent*. The same is true for the recording station. The output ranges are graphically displayed in the window at the bottom of the dialog. To turn on output for *Harmonic Analysis*, select the *Harmonic Analysis* item in the *Constituent*.

1. Enter the following values:
2. *Start Day* 24.0
3. *Output Every* 40 *Time Steps*
4. *End Day* 32.0
5. Click the *OK* button to return to the *Model Control* dialog..

4.2.6 Bottom Friction

Clicking on the Bottom Friction button brings up the Bottom Stress/Friction dialog. This dialog allows you to set how ADCIRC represents bottom stress. For this model, make sure *Constant* and *Quadratic* are selected with a coefficient of 0.0028. Then select OK.

4.2.7 Tidal Forces

One of the most powerful features of ADCIRC is the ability to use tidal forcing on the open boundaries of the grid. To define the tidal constituents that ADCIRC will apply at the ocean boundaries:

Tidal Potential Constituents

1. Click the *Tidal Forces* button from the main *Model Control* dialog.
2. To tell ADCIRC to run with tides, change the *Tidal Potential* to *On*.
3. Click the *New* button under *Tidal Potential Constituents*.
4. In the *New Constituent* dialog, make sure the *LeProvost* constituent database is selected. For this tutorial, you will be using the K1, O1, M2, S2 and N2 constituents.
5. Set the *Starting Day* as 0.00 hours on November 2, 1998 (0.00 hour, 2 day, 11 month, 1998 year). This is the date from which the tides will start.

- Click the *K1* constituent from the *Constituents* box on the right and click the *OK* button to return to the *Tidal Functions* dialog.
- Repeat steps 3 through 7 for the *O1*, *M2*, *S2*, and *N2* constituents.

Tidal Forcing Frequencies Constituents

Now, we have to tell *ADCIRC* to use these same constituents as forcing constituents on the open boundary of our finite-element mesh.

- Press the *Copy Potential Constituents* button. If a prompt appears that *SMS* cannot find *m2.legi*, push *OK* and find *m2.legi* in the file browser.
- SMS* takes each constituent and extracts the values it needs from the *LeProvost* constituent database, placing it into the *Tidal Forcing Frequencies Constituents* box on the right.
- Click the *OK* button twice to exit the *ADCIRC Model Control* dialog.

4.2.8 Solver

The *Solver* button brings up the *Solver* dialog which allows you to specify the final model parameters we will specify in this lesson. Set the following values:

Parameter	Value	Comment
Absolute Convergence	0.00001	
Max Iterations	25	
Solver Type	Iterative JCG	
Warning Messages	Fatal	

Click *OK* to leave the *Solver* dialog and *OK* again to leave the *Model Control* dialog.

4.2.9 Saving The Mesh and Control Files

To save the mesh and control files:

- Select *File | Save Project*.
- Enter the name “*shin_ready.spr*” and click the *Save* button.

The file is now ready to run in *ADCIRC*. The process could take between a few hours to over a week depending on the size of the grid and the processor being used. In the interest of time, you will not be running this model today. Later we will look at the results of a portion of a run. For now we will look at the other things you will want to know from *SMS*.

4.3 Editing a Mesh

Over time, you may find that you want to modify a mesh. This may be due to updated surveys, proposed channel realignments, extended application of the model which focuses on a different portion of the domain, or any of a number of other situations. The mesh may be edited manually by inserting and moving nodes, and swapping edges, but this is only sufficient for very minor changes. If you do perform changes on an existing mesh, it is a good idea to save the bathymetry of the mesh as a scatter set before beginning. If you have your original bathymetry data, this is not necessary. To save the mesh as a scatter set use the *Data | Mesh -> Scatter* command.

4.3.1 Replacing a Polygonal Area

Normally, when a mesh needs better resolution in a specific area, you should revise the size function used to create the mesh and regenerate the mesh. However, if you are working from an existing mesh that has no conceptual model, or you have made significant hand edits that would be lost by returning to the conceptual model, you may want to replace a portion of the mesh. Care must be taken to generate changes that will tie in to the existing mesh.

The easiest way to accomplish this is to actually delete the area you wish to replace. As an example, you may want to make the area offshore from the inlet have a higher density. Select the nodes in the area you want to replace. You can do this using the *Select by Polygon* command in the *Edit Menu* with the *Select Node* tool active.

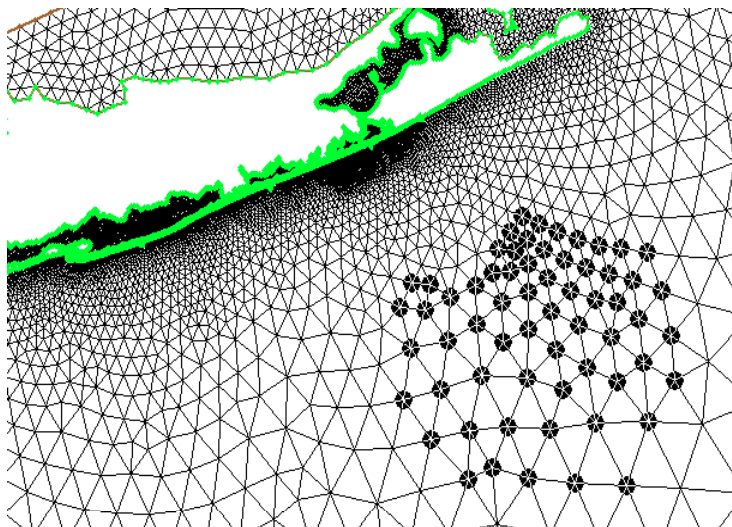


Figure 4-1 Nodes selected to be replaced

Delete these selected nodes creating a hole in the mesh. If you have any really jagged edges to the hole, select the nodes at the points of the jagged edges and delete them too.

Now we need to create a polygon that matches the hole. To do this:

1. Select the *Create Nodestring* tool.
2. Click on any node on the edge of the hole.
3. Hold down the CTRL key and select the node counter clockwise around the hole.
4. Double click on the first node you selected. You now have a nodestring that follows the edge of the hole.
5. Now select a layer of elements around the edge of the hole. Delete these too, and generate another nodestring around the new edge. This area will be the transition zone from the existing resolution to the new resolution.

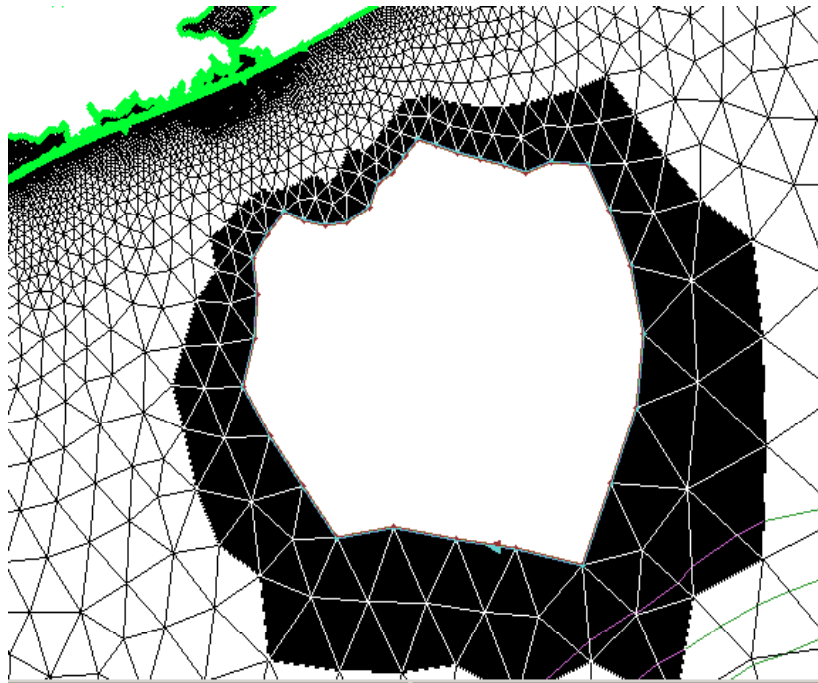


Figure 4-2 Nodes selected around edge of original hole.

Now we want to fill the hole with elements. To do this, we want to create a conceptual model of the hole. To do this:

1. Select the two nodestrings (one around the hole, and one in the hole).
2. Use the *Data | Mesh -> Map* command to create arcs along these nodestrings.
3. In the convert dialog, select the *Nodestrings->Arcs* option.

Now you have the arcs, switch to the *Map* module, set the coverage type to *ADCIRC*, and build polygons (*Feature Objects | Build Polygons*).

Using the *Select Polygon* tool, double click on the area inside the inner arc, and adjust the polygon attributes. You can change the distribution of vertices along the inner arc, you can split it into multiple arcs and adjust the bias. Make sure you leave the outer arc alone. It cannot be edited, or the new elements will not interface cleanly with the existing mesh.

When you have the polygon option set:

1. Select the *Data | Map-> 2D Mesh* command.
2. Make sure the *Delete existing mesh* toggle is NOT selected.
3. Click Ok.

A new set of elements are created to fill in the hole.

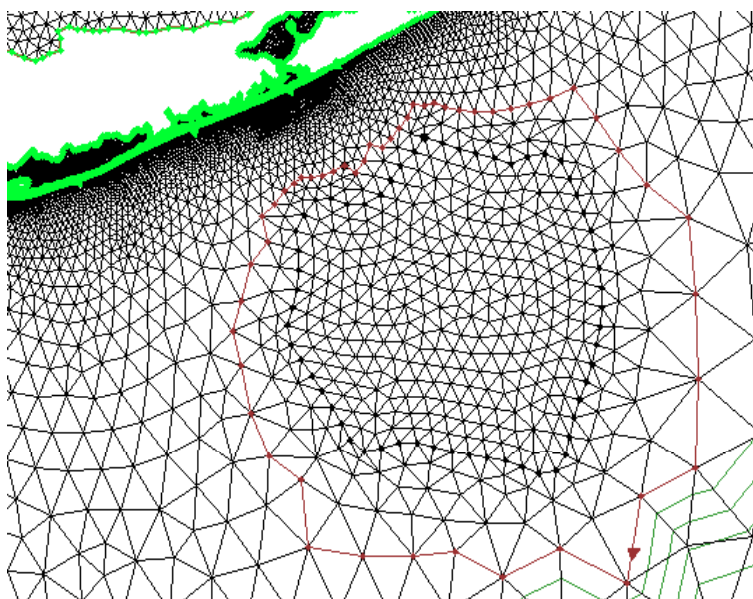


Figure 4-3Denser mesh fit into existing mesh.

You will want to find and remove disjoint nodes in the mesh and renumber the mesh before saving. This example only serves as an illustration of the method. You will not be asked to save this change.

4.3.2 Reinterpolating Bathymetry

After editing a mesh, if you have moved nodes or replaced nodes that don't have the appropriate elevations associated. Or if you have a new survey that you want to use to update the bathymetry of an existing mesh, you can reinterpolate the elevation values from a scattered data set to the mesh. To do this:

1. Go to the *Scatter Point Module*


2. Select the scatter point set you want to use as a source of elevation data.
3. Select *Interpolate | To Mesh....* This will bring up the *Interpolation Options* dialog.
4. Select the data set that defines the new bathymetry.
5. Turn on the *Map to Z Value* toggle..
6. Select *OK* to perform the interpolation.

This method can be used to interpolate other data to a mesh (without mapping the Z values) to visualize other quantities on the mesh domain.

4.4 Running ADCIRC

Here are a few final notes on running *ADCIRC*. Presently, *ADCIRC* uses a specific naming convention for its input and output files. Therefore, before *ADCIRC* can start, the basic input files must be present in the working directory, which *SMS* does automatically. *SMS* makes a copy of the active mesh file and names it “fort.14,” then makes a copy of the model control information file and names it “fort.15.” The *ADCIRC* executable also needs to be located in the directory where the files are located.

To run *ADCIRC*:

1. Move or copy the *ADCIRC* executable into the directory where you saved your project files.
2. Select *ADCIRC | Run ADCIRC*.
3. If the name of the *ADCIRC* executable does not appear, click the folder icon , locate the *ADCIRC* executable, and click OK. (Note: this takes too long to perform the analysis during this interactive lesson. If you want to start the executable to see it run, you can, but you will want to abort the run.)

Once the *ADCIRC* run has completed, there will be several newly created files. *SMS* copied the “shin_ready.grd” file (the mesh file saved when the project file was saved) to “fort.14” and “shin_ready.ctl” file to “fort.15,” the filenames needed by *ADCIRC*. *ADCIRC* created the “fort.63” (global elevation) and the “fort.64” (global velocity) files. There are a couple of other files that hold basic output information, but we will only focus on the elevation and velocity files for the remainder of this tutorial.

4.5 Conclusion

NOTE: *ADCIRC* may take several hours or even days to run. Press CTL+C to stop *ADCIRC*. A completed run has been made and those files will be used in the next Lesson.

This concludes the *ADCIRC Mesh Editing* workshop. You should now be familiar with some of the features that *SMS* provides for editing the finite element mesh. You may continue to experiment with the interface or you may exit the program.

If you wish to exit *SMS* at this point:

1. Choose *File | Exit*. Click the *Yes* button if asked to confirm.